GEOMETRIC MODELING FOR FINITE ELEMENT USING MULTI-REGIONS AND PARAMETRIC SURFACES

W.W.M. Lira[a], **A.C.O. Miranda**[a],, **L.C.G. Coelho**[a], **J.B. Cavalcante Neto**[b,c] and **L.F. Martha**[a]

[a]Department of Civil Engineering and Computer Graphics Technology Group (Tecgraf),
Pontifical Catholic University of Rio de Janeiro (PUC-Rio) 22453-900,
{william,amiranda,lula,lfm}@tecgraf.puc-rio.br

[b]Cornell Fracture Group,
Cornell University, USA.
joaquimb@vienna.cfg.cornell.edu

[c]Computer Graphics Group,
Federal University of Ceara (UFC), Brazil.
joaquimb@lia.ufc.br

This work presents a computational environment for geometric modeling applied to finite-element analysis using multi-regions and parametric surfaces represented by NURBS. This environment is based on the integration of computational techniques and methodologies previously presented by the authors. The main goal is the generation of 3D models to be used in numerical simulations based on the Finite-Element Method. For this purpose, the adopted methodology consists of combining aspects of geometric modeling, such as surface intersection and automatic region recognition, with solid finite-element mesh generation in arbitrary domains.

Geometric modeling is supported by a hybrid data representation scheme [1] that allows the detection of closed-off solid regions described by surface patches. These patches may either be interactively created by means of an efficient graphics interface or may result from parametric-surface intersection. The surface intersection [2,3] integrates the problem of parametric-surface intersection with finite-element surface mesh compatibility at intersection curves. Finite-element surface mesh generation is performed in parametric space and combines a quadtree-based refinement with an advancing-front technique for the generation of an unstructured mesh [4]. Finite-element volume mesh generation uses the same concepts but an octree instead of the quadtree [5].

In the proposed environment, simulation attributes are managed by a system called ESAM (Extensible System Attributes Management), which allows the customization of simulation attributes. The resulting modeler can be extended to different types of engineering problems. Examples of geometric and FEM modeling of realistic engineering problems are presented to validate the proposed computational procedures.

References

[1] W.W.M. Lira, P.R. Cavalcanti, L.C.G. Coelho, and L.F. Martha, “A Modeling Methodology for Finite Element Mesh Generation of Multi-Region Models with Parametric Surfaces”, *Computers & Graphics*, v. 16, n. 6, 2002 (In Press).

[2] W.W.M. Lira, L.C.G. Coelho, and L.F. Martha, “Multiple Intersections of Finite-Element Surface Meshes”, *Proceedings of 11th International Meshing Roundtable*, pp. 355-363, 2002.

[3] L.C.G. Coelho, M. Gattass, and L.H. Figueiredo, “Intersecting and Trimming Parametric Meshes on Finite-Element Shells”, *International Journal for Numerical Methods in Engineering*, v. 47, pp. 777-800, 2000.

[4] A.C.O. Miranda and L.F. Martha, “Mesh Generation on High-Curvature Surfaces based on a Background Quadtree Structure”, *Proceedings of 11th International Meshing Roundtable*, pp. 333-341, 2002.

[5] J.B. Cavalcante Neto, P.A. Wawrzynek, M.T.M. Carvalho, L.F. Martha, and A.R. Ingraffea, “An Algorithm for Three-dimensional Mesh Generation for Arbitrary Regions with Cracks”, *Engineering with Computers*, v. 17, n. 1, pp. 75-91, 2001.